Thermal Protection System Cavity Heating for Simplified and Actual Geometries using Computational Fluid Dynamics Simulations with Unstructured Grids

Peter L. McCloud[1]
The Boeing Company, Houston, Texas, 77571

Thermal Protection System (TPS) Cavity Heating is predicted using Computational Fluid Dynamics (CFD) on unstructured grids for both simplified cavities and actual cavity geometries. Validation was performed using comparisons to wind tunnel experimental results and CFD predictions using structured grids. Full-scale predictions were made for simplified and actual geometry configurations on the Space Shuttle Orbiter in a mission support timeframe.

I. Introduction

In the wake of the Columbia Accident, a large scale effort was made by the Aerothermodynamics community to further understand the nature of heating due to off nominal TPS conditions (protrusions, cavities, breaches, etc). As part of this effort Computational Fluid Dynamics (CFD) has become a key tool in providing heating. To support Space Shuttle missions, a CFD repository has been created using the state of the art in Aerothermodynamics codes, DPLR[1] and LAURA[2], both of which use structured grids. As part of the Aeroheating mission support, CFD is used real time during the mission to evaluate TPS damages[3].

Part of this support involves generating Bump Factor (BF) distributions for cavity damages. The traditional methodology is to use a patch grid method. A localized grid is generated containing a simplification of the damage site and the surrounding Outer Mold Line (OML). The flow field conditions at the outer boundary of the grid are fixed to the flow field conditions of the smooth OML vehicle from the CFD Repository. During a typical mission once cavity damage has been defined the CFD teams perform gridding, run the solution and perform post processing in a 12 to 24 hour time frame. A large scale validation effort[4] has been used to verify the Cavity Heating CFD results used for mission support.

Recently an effort was undertaken at Boeing Houston to predict cavity heating using unstructured grids with the end goal of providing cavity heating for actual cavity geometries in a time frame useful for mission support. Unstructured gridding has a strong heritage, but unstructured hypersonic solvers are still an emerging technology. At the fore front of unstructured hypersonic solver development is Gnoffo et al[4] and the HEFSS team and Chandler et al[5] with the US3D code. However the real technology drivers for these codes are to provide solutions for an entire vehicle on an unstructured grid. By continuing to use the patch methodology, the unstructured grid domain lies away from the stagnation region and away from the shock and the requirements on the solver are more benign.

II. Solver Overview

For the current task Loci-CHEM[6] (hereby referred to as CHEM) was used to solve the patch grid and the flow conditions on the outer boundary were fixed by the DPLR CFD Smooth OML solutions. CHEM is finite volume based flow solver developed using the Loci[6] framework, both of which have been developed by Dr. Ed Luke at Mississippi State University (MSU). The code works with unstructured grids with arbitrary cells. The solver is written so that add-on modules can be easily be developed in the Loci framework and incorporated into an analysis.

[1] Entry Aero/Aeroheating Engineer, Space Exploration – Space Shuttle Program, MC HB2-30, Member, AIAA

The presence of a strong chemistry formulation, code adaptability, and existence of companion conduction and radiation solvers made CHEM from the standpoint of the Boeing Houston personal, a good candidate for this task.

Cons for CHEM for performing entry aerothermodynamics analysis is that it does not contain either a radiation adiabatic boundary condition or a catalytic wall boundary condition was available. Additionally the existing air models are limited to 5000 K and there also is not a thermodynamic non-equilibrium model, but again by staying away from the stagnation region with the cavity patches, there is not a need for either of these features.

III. Validation with Simplified Cavities

The initial portion of the project was the perform validation. As part of the Return to Flight (RTF) support after the Columbia accident a large number of runs at the Langley Mach 10 wind tunnel were made in support of cavity heating. For these runs cavities of various geometries were machined into plates and inclined at various angle of attacks to simulated different flow conditions. A subset of these runs was used to validate the DPLR and LAURA cavity heating predictions. To validate the CHEM cavity predictions the key runs used to validate DPLR and LAURA were used to compare against. Due to time constraints the same level of effort could not be made as the prior validation efforts.

A patch method was used to model the wind tunnel runs, with the outer boundary of the cavity patch grid set to the flow conditions predicted by DPLR for a smooth OML plate. CHEM was run once with a hexahedral grid directly imported from the DPLR solution and again with an unstructured grid.

A. Short Cavity Results

The test article for the test series T406, run 45 consisted of a short, Everhart cavity with a L/H of 7.1 machined into a test plate. Overall comparisons for both the hexahedral and the unstructured grids show that CHEM is capturing the heating in the cavity.

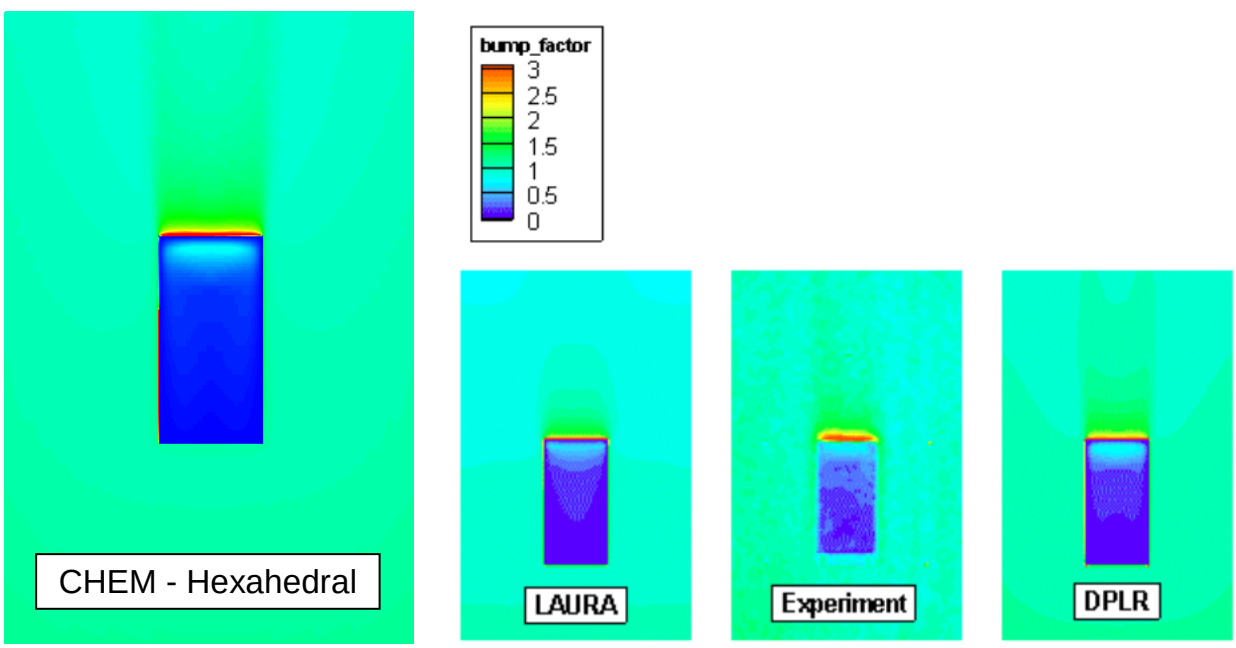


Figure 1. Comparisons of Cavity Heating of CHEM Hexahedral solution vs. DPLR/LAURA and Experiment.

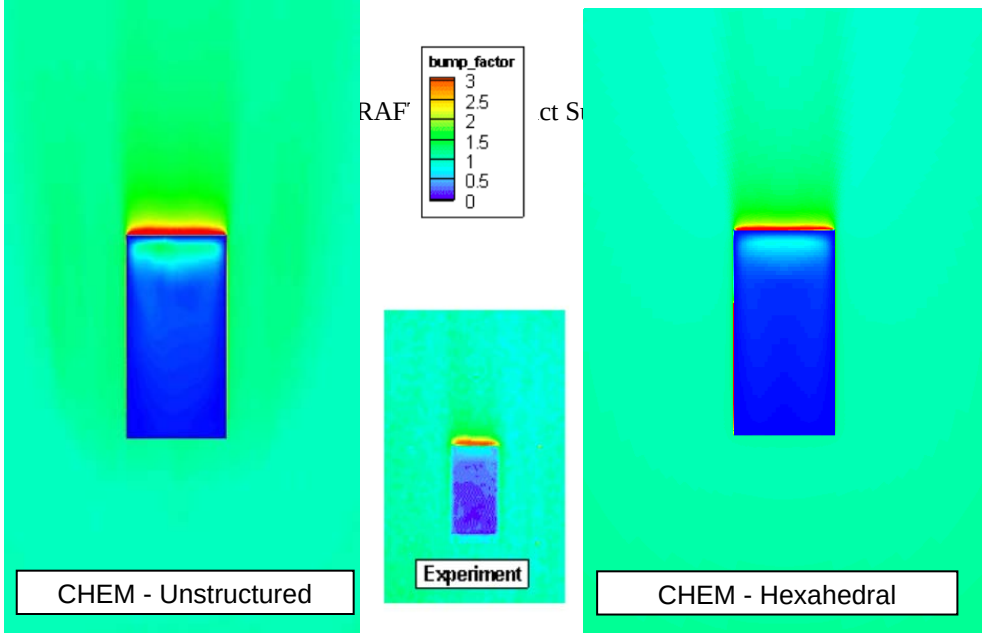


Figure 2. Comparisons of Cavity Heating of CHEM Unstructured solution vs. CHEM Hexahedral and Experiment.

B. Long Cavity Results

Run 35 from the T406 Test series consisted of a long cavity with an L/D of 19.0 machined into a flat plate. The long cavities are more difficult computationally as the flow impinges on the cavity floor and vortices are generated along the side walls. The differences between CHEM and the other solvers are more pronounced for these cases, but overall CHEM is capturing the cavity heating. The vortex heating on the midway down the length of the cavity floor is higher for the unstructured grid than for the unstructured grid

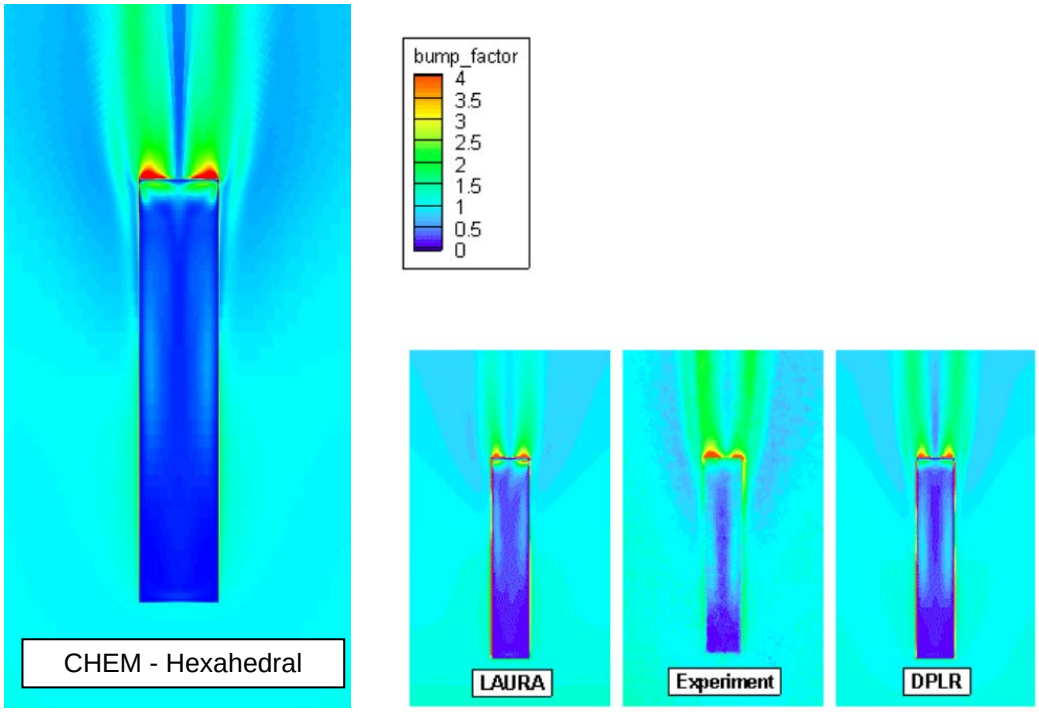


Figure 3. Comparisons of Cavity Heating of CHEM Hexahedral solution vs. DPLR/LAURA and Experiment.

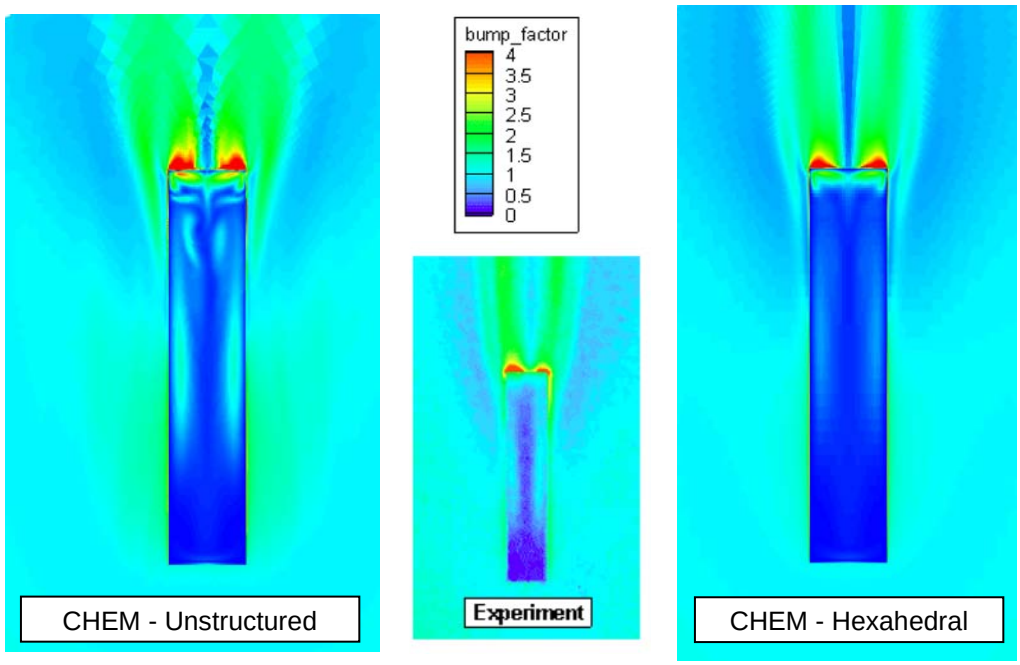


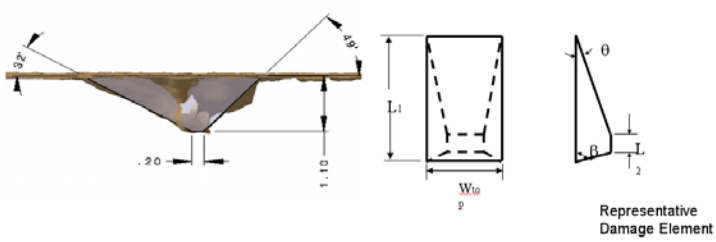
Figure 4. Comparisons of Cavity Heating of CHEM Unstructured solution vs. CHEM Hexahedral and Experiment.

IV. Actual Cavity Geometries

With confidence for the ability of CHEM to predict Cavity Heating established for simplified cavities, the project moved forward in working with actual cavity geometries. During a typical shuttle mission an initial damage inspection is performed by taking photography during the R-bar Pitch Maneuver (RPM). If a damage site requires additional information to clear the site or for a repair, a Focused Inspection (FI) is performed and typically for cavity damages a high fidelity point cloud is generated from a laser scan. This point cloud can then be used to generate an actual cavity geometry definition to build an unstructured grid.

A. STS-118 600_2-001

During STS-118 a lot of time was effort was spent to clear the 600_2-001 damage site. A couple of different simplified cavities were defined to fully understand the flow conditions and the cavity heating. With the wealth of information available for this damage site, it was chosen as one of the case studies.



OBSS ID	L_1 (in)	W_{top} (in)	D_1 (in)	L_2 (in)	W_{bottom} (in)	θ (deg)	β (deg)	α (deg)	γ (deg)	VOLUME (in^3)
M-118-RPM-600_2-001-Case2	2.92	1.40	1.10	0.20	0.70	32.0	49.0	78.0	67.0	2.0

Figure 5 - STS 118 600_2-001 Simplified Cavity 2 Definition

One of the simplified cavities, #2, was analyzed using CHEM and compared to the DPLR and LAURA results to ensure consistency and to make sure that the boundary conditions were correct. Results showed that while the recirculation zone is in a slightly different location, overall the cavity heating is being predicted correctly for the Flight Conditions.

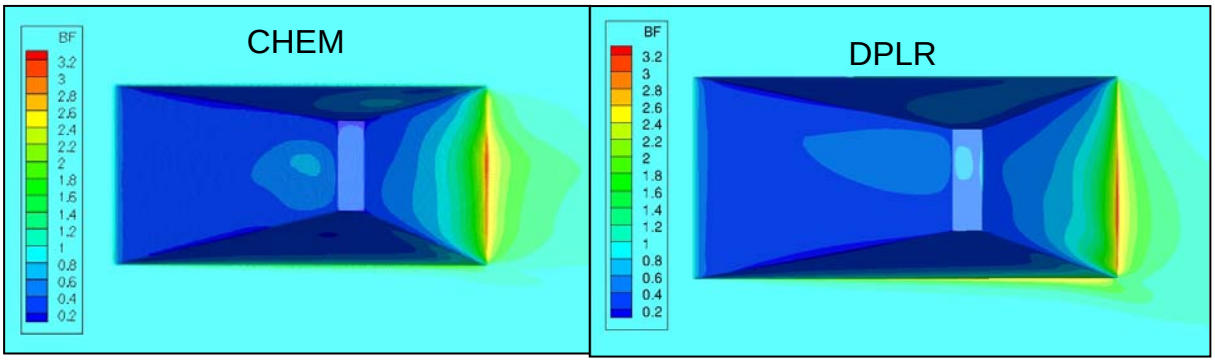
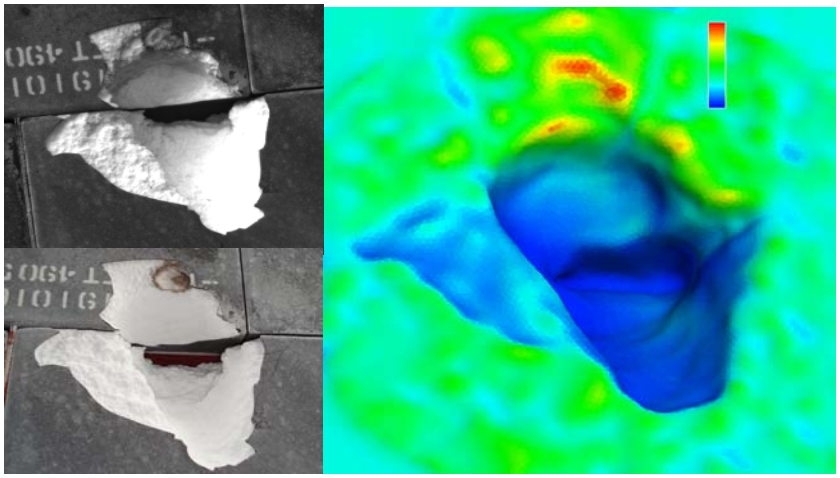


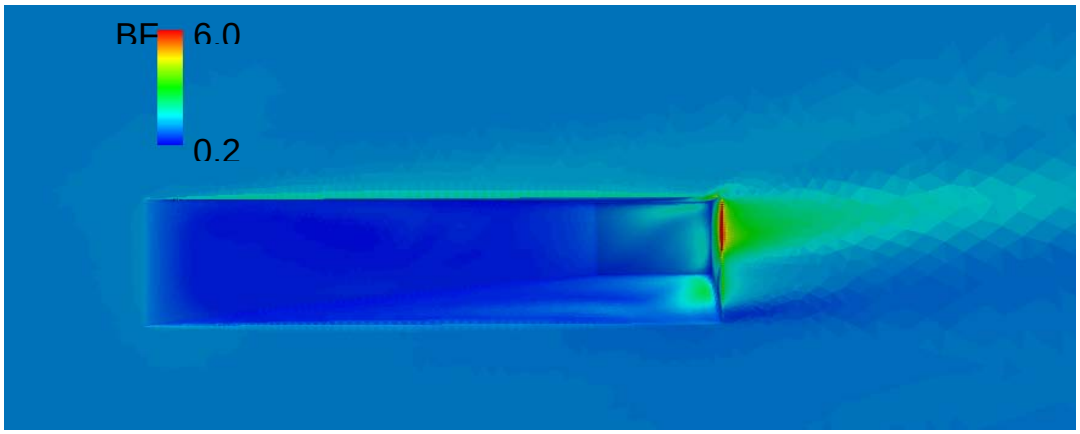
Figure 6 – Cavity Heating for STS 118 600_2-001 Simplified Cavity 2

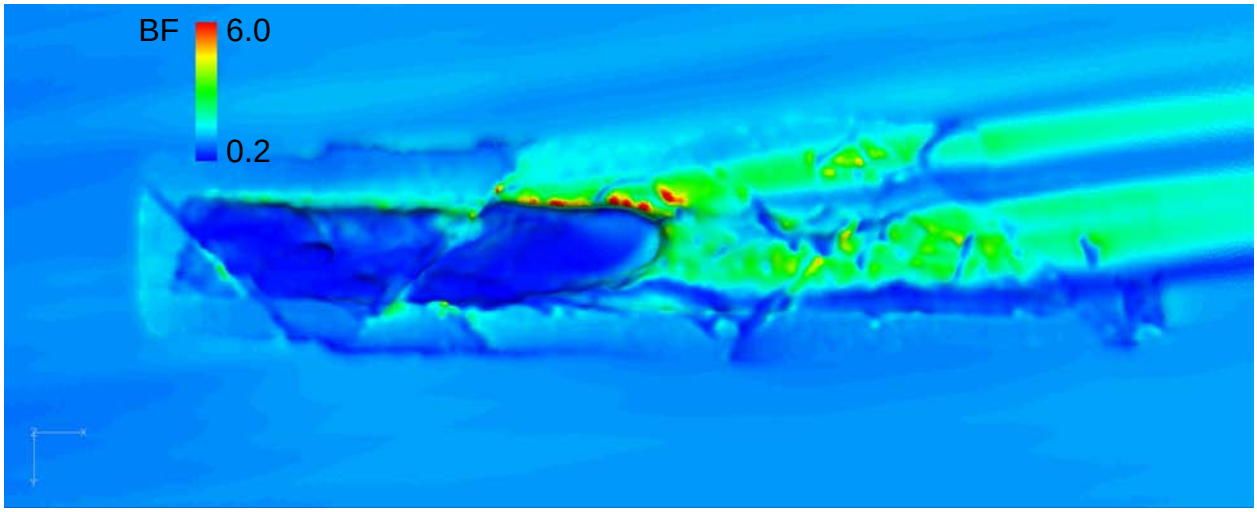
The actual geometry configuration was run using CHEM. The simulation with the actual geometry supports the conclusion from the simplified cavities that the BF at the floor of the cavity is low. The actual cavity geometry predicts a BF of 0.25 is appropriate, vs. 0.47-0.53 with the actual geometries, so the simplified cavities were conservative.



B. STS-133 Simulation

A simulation was run in September of 2010 in preparation for STS-133. One of the simulated damages was a large cavity damage in the tile acreage. Both a simplified and actual cavity geometry was generated to understand the cavity heating.





The ranges of heating factors for both the simplified and actual geometry cavities are the same. What is most interesting is 1) the shift of the peak heating location from the rear wall to the side wall and 2) the increased length of the heating aft of the cavity.

V. Conclusion

The use of unstructured grids for cavity heating factors can be done by employed a patch grid method with smooth OML solutions provided by solvers such as DPLR. Simulation of the actual cavity geometry can provide additional cavity heating information for complex geometries.

References

[1]Wright, M. J., Candler, G. V., and Bose, D., "Data-Parallel Line Relaxation Method for the Navier–Stokes Equations," AIAA Journal, Vol. 36, No. 9, 1998, pp. 1603–1609.

[2]Gnoffo, P. A., "An Upwind-Biased, Point-Implicit Algorithm for Viscous, Compressible Perfect-Gas Flows," NASA TP-2953, Feb. 1990.

[3]Tang, C., Saunders, D., Trumble, K., and Driver, D., "Rapid Aerothermal Simulations of Damage and Repair during a Space Shuttle Mission," AIAA Paper 2007-1783, April 2007.

4Gnoffo, P. A., "Updates to Multi-Dimensional Flux Reconstruction for Hypersonic Simulations on Tetrahedral Grids", AIAA Paper 2010-1271, 48th AIAA Aerospace Sciences Meeting, Orlando, Florida, January 4-7th 2010.

[5]Nompelis, I. N., Drayna, T., and Candler, G. V., A Parallel Unstructured Implicit Solver for Hypersonic Reacting Flow Simulations, AIAA Paper 2005-4867.

[6]Edward Luke, "On Robust and Accurate Arbitrary Polytope CFD Solvers" AIAA-2007-3956 18th AIAA Computational Fluid Dynamics Conference, Miami, Florida, June 25-28, 2007